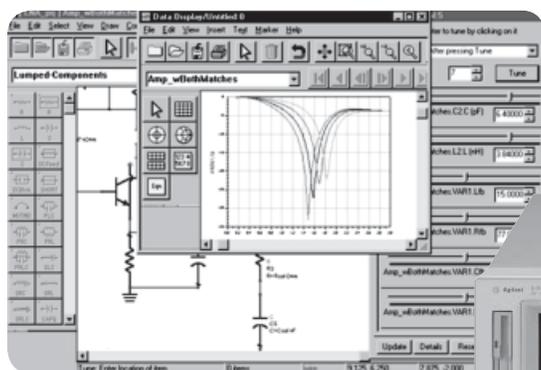


Agilent

Achieving Fast Design Cycle Time Using an Electronic Design Automation (EDA) Tool and Impedance Analyzers

Application Note 1369-5



Introduction

Higher performance, smaller physical size, reduced manufacturing cost, and, decreased cycle time represents some of the strongest trends in product design today. One method of achieving fast cycle time is to simulate the characteristics of the circuit before the circuit design is actually in layout on the printed-circuit board. By simulating with EDA tools, the circuit design can be verified and corrected without having to spend time waiting for the re-designed circuit to be fabricated again.

As a result, development phases of circuit blocks have shrunk and this short development phase matches the ever-changing needs and requirements of the wireless communication equipment, which typically have shelf lives that are less than a year.

Electronic component manufacturers, whose products range from filters to amplifiers, have also started to design their prototypes using EDA tools. Recent developments in EDA tools make it possible to optimize component designs using real world tradeoffs such as Q (quality factor), impedance, phase noise, cut-off frequency, cost, etc.

In this product note, it is explored how fast cycle time is achieved in the following two areas by using the E4991A RF Impedance/Material Analyzer and an EDA tool:

- Equivalent circuit modeling of passive components in the design phase of the component manufacturer R&D engineer.
- Effective selection of passive components in the design phase of the equipment manufacturer circuit designer.

Information on using the 4294A precision impedance analyzer for device modeling is given in Appendix 2 at the end of this application note.



Agilent Technologies

Agilent E4991A Features

Superior impedance measurement accuracy

The E4991A RF impedance/material analyzer provides direct readings of impedance measurements at frequencies from 1 MHz to 3 GHz and impedances from milliohms to kilo-ohms, making it ideal for the characterization of components such as capacitors, inductors, diodes, and EMI filters. The E4991A employs the RF I-V measurement technique to achieve a basic measurement accuracy of 0.8%. The RF I-V method measures impedance directly from a ratio of voltage and current, enabling high measurement repeatability over a wide frequency and impedance range (Figure 1).

SMD test fixtures provide excellent measurement repeatability

The latest passive components are smaller and lighter, have higher Q and stricter tolerances. In accordance with this trend, it is becoming extremely difficult for users to evaluate passive surface-mount devices (SMD) with high accuracy and repeatability. In order to solve these problems, various SMD test fixtures are available for the E4991A.

The SMD test fixtures 16197A and 16196A/B/C/D provide highly repeatable measurement results in the range of 1 MHz to 3 GHz.

Linkage with EDA tools

By using the CITIFILE format, impedance characteristics measured using the E4991A and an SMD test fixture, can be imported into an EDA tool. Measurement data taken by E4991A is first converted into S-parameters and then saved as a CITIFILE (See Figure 3).

The CITIFILE can then be transferred to the PC/workstation by using either the GPIB or LAN interfaces or by saving it onto a floppy disk. When using the LAN interface, the CITIFILE can be transferred by using FTP or the remote user interface. Once the CITIFILE is transferred to the PC/workstation, it can be imported into an EDA tool¹.

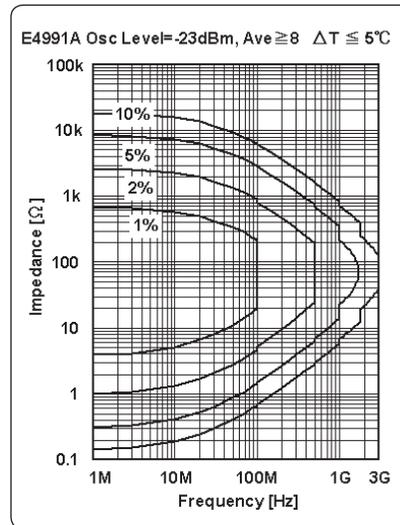


Figure 1. Impedance measurement range of E4991A



Figure 2. 16196A/B/C/D and 16197A

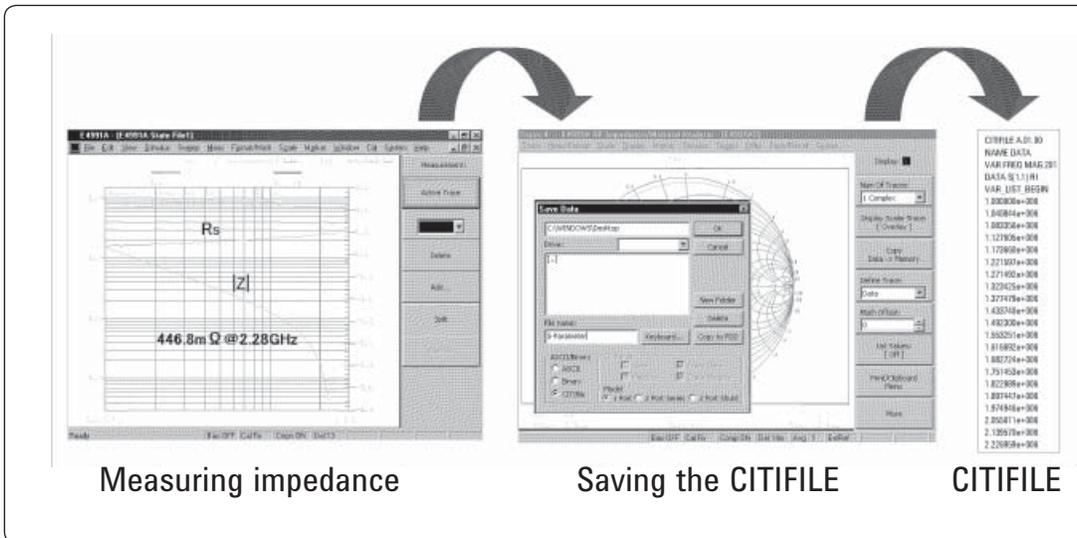


Figure 3. Creating the CITIFILE

1. As for the method of importing the CITIFILE into your EDA tool, please refer to the respective user manual.

Equivalent Circuit Modeling of Passive Components

Design phase of the component manufacturer R&D engineer

A component manufacturer R&D engineer's responsibility is to design a high quality component in a short period of time. Given the present environment of the market's ever-changing needs and the short shelf life of wireless communication equipment, a fast cycle design time is a prerequisite in today's industries. Once the component is developed it must be transferred to manufacturing for mass production and the characteristics must be optimized to a tolerance and documented so that the component can be utilized by end-users to the fullest extent.

Here the engineer's most common design phases are illustrated in Figure 4, using an LCR (inductor, capacitor or resistor) component as an example. The LCR component is designed using an EM¹ simulator to determine the initial structure and material composition. The prototype LCR component is then built to specifications. Using an impedance analyzer the characteristics are verified and the equivalent circuit model is simulated to help with the re-design of the prototype. Once the design of the LCR component is complete, the documentation begins with typical results recorded in application notes for the end-user. S-parameter library data and equivalent circuit models are also prepared so that end-users can use them to select the appropriate device for their circuit design.

Problems in the characterizing the LCR component

The primary issue facing the component manufacturer R&D engineer in the early development phases is the equivalent circuit modeling of the passive component. Impedance and network analyzers can only measure and display the impedance of components based on one complex parameter, using the series resistance (Rs) and reactance (Xs) elements for a series model and the conductance (G) and susceptance (B) for a parallel model.

Such simplified models can be used to derive parameters such as Lp, Ls, Cp, Cs, Rs, Rp, D, or Q, but this is not enough to completely characterize the frequency response of the component's equivalent circuit.

Impedance analyzers such as the E4991A have an equivalent circuit analysis function to extract parameters of the five most common equivalent circuit models (See Figure 5). However, these models are limited to the number of elements that can be accommodated, resulting in an inaccurate analysis for high frequencies.

Using the E4991A and EDA tools for a fully characterized equivalent circuit model

Many component manufacturer R&D engineers have started to use EDA tools for the equivalent circuit modeling of passive components. Methodologies range from SPICE extractions to best curve-fit. No single methodology can be classified as best. Still, the advantage is that an accurate model with an optimum number of elements can be created.

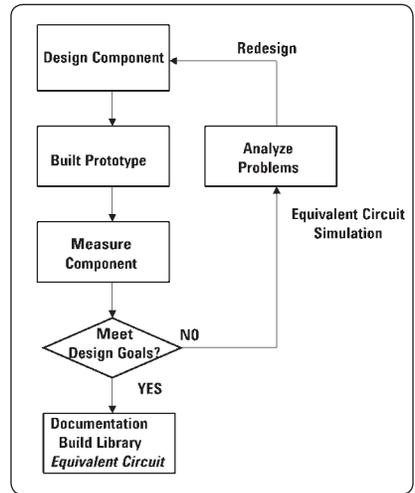


Figure 4. Design phase of the component manufacturer R&D engineer

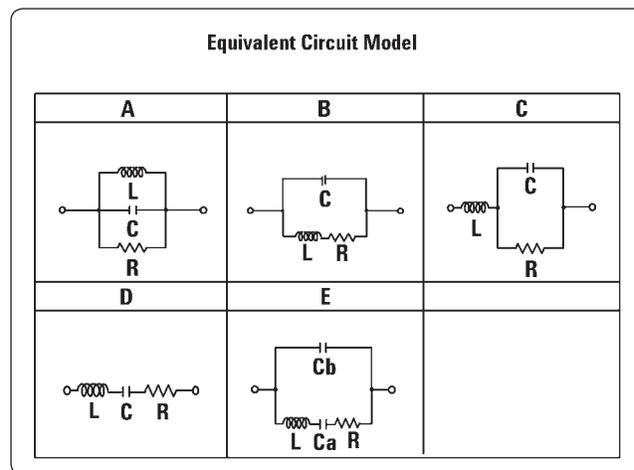


Figure 5. Equivalent circuit models in E4991A

1. Electromagnetic and 3-dimensional planar analysis.

Equivalent circuit modeling of an inductor

Here, a simple method, which involves the Agilent EEs of Advanced Design System (ADS) and the E4991A, is introduced. This method has the advantage of employing just one goal over a wide frequency range.

The flowchart in Figure 6 shows the steps that should be taken for this equivalent circuit simulation. Each step is described here:

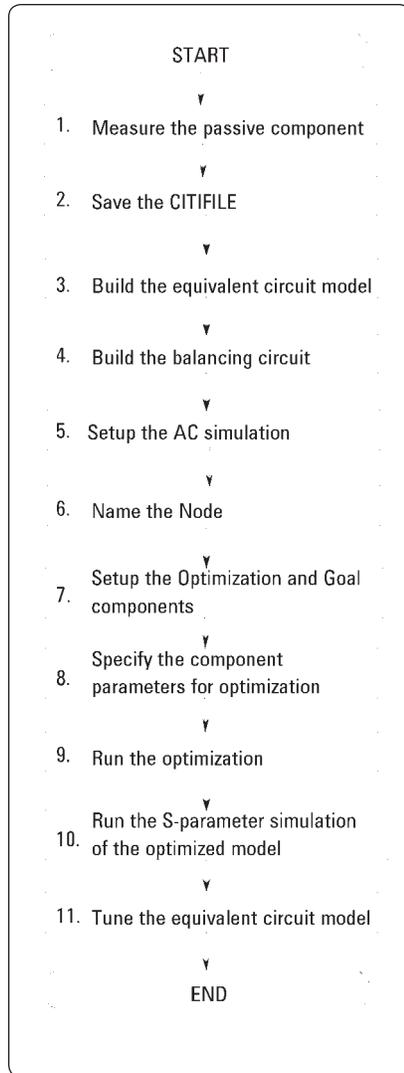


Figure 6. Flow chart for the equivalent circuit simulation procedure

Step 1. Measure the passive component

Measure the passive component using the E4991A and either the 16196A/B/C/D or the 16197A. It is a good idea to use a high averaging factor to reduce the effects of trace noise.

Step 2. Save the CITIFILE

Save the measured data as a CITIFILE with 1-port S-parameters and import the CITIFILE into ADS using the Data Item component¹.

Step 3. Build the equivalent circuit model

Build the equivalent circuit model of the passive component. If the basic model is not known, the model can be decided upon by doing a few simulations using the equivalent circuit analysis function in the E4991A. By doing a few simulations, one can decide which model A through E is close to the component's characteristics. Once the model is chosen, additional elements can be added in the ADS schematic for a better fit for high and low frequencies (See Figure 7). In this example, since the basic model was known, the equivalent circuit model shown in Figure 8 was used.

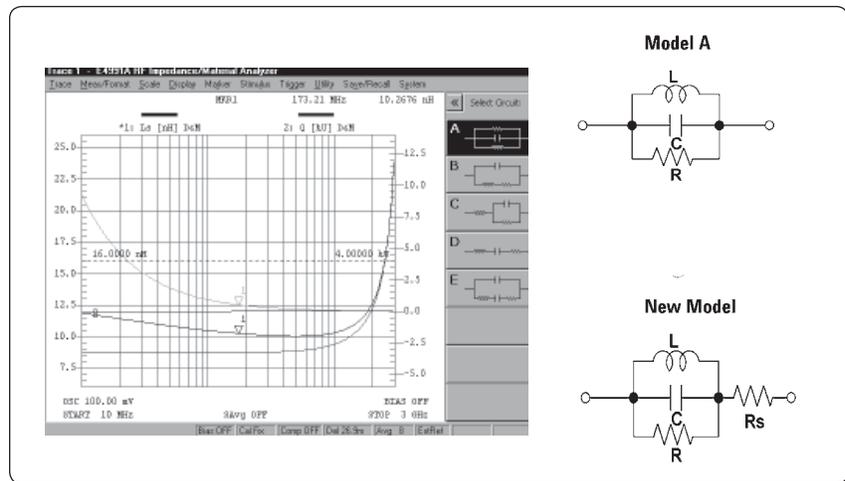


Figure 7. Build the equivalent circuit model using E4991A

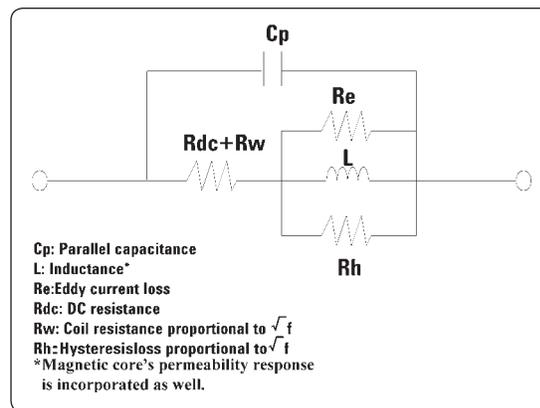


Figure 8. Equivalent circuit model for an inductor with a magnetic core

1. Allows measurement data to be used in the simulation.

Step 4. Build the balancing circuit

The “balancing” circuit as shown in Figure 9, balances the voltages applied to the equivalent circuit model and the Data Item component. This is accomplished by applying two signals that are 180 degrees out of phase to each other and optimizing the node voltage between the equivalent circuit model and the Data Item component to be close to zero. As a result of the optimization, each element in the equivalent circuit, which is pre-defined as a variable, is automatically set to some value.

Symbolically, this can be represented as a least-squares equation, where Z is the set of impedance measurement data taken by the E4991A and E_{qvc} is the set of possible solutions to the equivalent circuit.

Figure 10 shows the balancing circuit created for a 47nH inductor.

$$\sum |Z(f_i) - E_{qvc}(f_i)|^2 \rightarrow 0$$

Step 5. Setup the AC simulation

Define the Simulation-AC component¹ in the ADS schematic in order to have the balancing circuit operate with an AC sweep over a prescribed frequency range.

Step 6. Name the node

Name the node between the equivalent circuit model and the Data Item component in the ADS schematic.

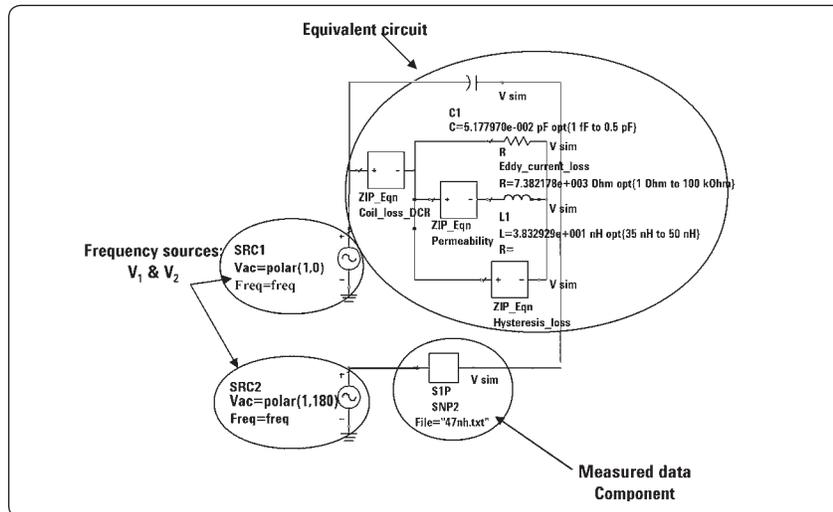


Figure 10. Balancing circuit for a 47nH inductor

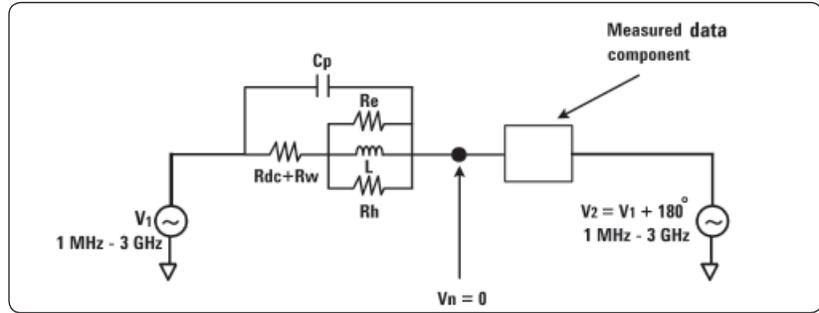


Figure 9. Balancing circuit

Step 7. Setup the Optimization and Goal components

Define the Nominal Optimization and Goal components in the ADS schematic. In the Nominal Optimization component, the appropriate optimizer, number of iterations, and the appropriate goal item should be selected. In the Goal component, the magnitude of the node voltage should be limited to be between 0 and 0.001 and the frequency range where the node voltage should be limited is set (Figure 11).

NOMINAL OPTIMIZATION

Optim
 Optim1
 OptimType=Random
 ErrorForm=MML1
 MaxIters=100
 P=2
 DesiredError=0.0
 StatusLevel=4
 SetBestValues=yes
 Seed=5
 SaveSols=yes
 SaveOptimVars=no
 SaveGoals=no
 GoalName[1]="OptimGoal1"

GOAL

OptimGoal1
 Expr="mag(Vnode)"
 SimInstanceName="AC1"
 Min=0
 Max=0.001
 Weight=
 RangeVar[1]="freq"
 RangeMin[1]=1GHz
 RangeMax[1]=3GHz

Figure 11. Optimization and Goal components in ADS

1. Performs a swept frequency or swept variable small-signal linear AC simulation.

Step 8. Specify the component parameters for optimization

In order for the goal to be achieved, before running the optimization, each of the elements in the equivalent circuit must be defined as variables that can be optimized. Each of the elements contains parameters that can be optimized over a pre-selected parameter range. Enable the optimization status of only relevant parameters and select as narrow a parameter range as possible. Pick initial values obtained from the equivalent circuit simulation performed with the E4991A firmware or from the EM analysis.

Step 9. Run the optimization

Run the optimization and after the AC simulation is completed, check that the node voltage is close to 0. Figure 12 shows the node voltage plot for the simulation performed using balancing circuit of Figure 10.

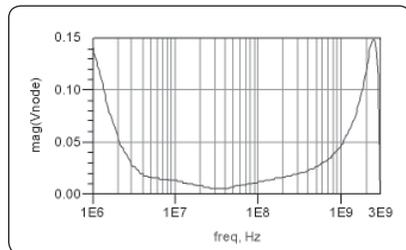


Figure 12. Optimized node voltage plot

Step 10. Run the S-parameter simulation of the optimized model

Copy the optimized equivalent circuit model into a S-parameter simulation template and construct a 1-port configuration. Run the S-parameter simulation and use equations to convert the simulation results into impedance parameters such as L_p , L_s , C_p , C_s , R_s , R_p , D , or Q . Display the (converted) simulation results with the measured impedance data and compare them.

Figure 13 shows the comparison of the measured impedance data of a 47nH inductor and the simulation results. Notice that simulated L_s and Q frequency response is close to the measured data.

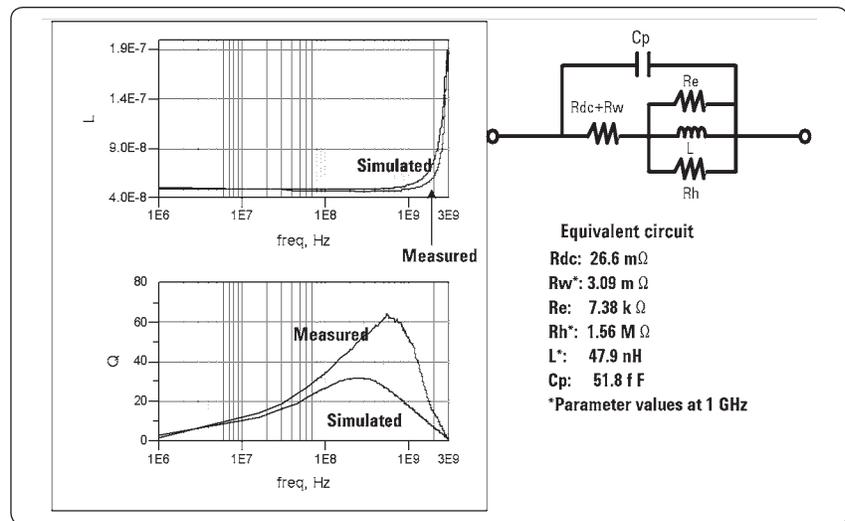


Figure 13. Simulation results of the optimized model

Step 11. Tune the equivalent circuit model

Use the tuning function in ADS to tune the equivalent circuit's frequency response to better match the measured impedance data.

Figure 14 shows the comparison after the tuning is completed. Notice that the simulated L_s and Q frequency response is much closer to the measured impedance data.

As a result of this equivalent circuit simulation, each of the loss and storage elements of the inductor were quantified. As a result, the component manufacturer R&D engineer can analyze and redesign the inductor.

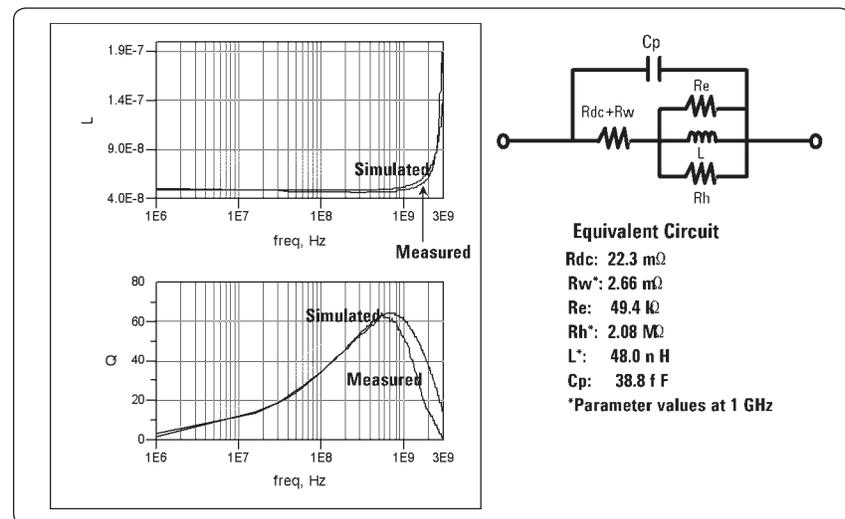


Figure 14. Simulation results of the optimized model after tuning

Effective Selection of Passive Components

Design phase of the equipment manufacturer circuit designer

A circuit designer's responsibility is to design a circuit block for an electronic device in a short period of time.

Here the circuit designer's development phases are illustrated using a wireless handset as an example (See Figure 15). The wireless handset's specification goals are first mapped out in R&D and product planning. The circuit designer roughly designs the RF circuit block (to be used in the wireless handset) with available components using an EDA tool. After this step, components are selected for the final design with respect to the planned budget and specifications goals of the circuit block. In the selection process of the components, measurements of the components are made with a network analyzer and used in the circuit simulation. S-parameter data provided by the component's manufacturer may be utilized for the circuit simulation.

The component that yields the best characteristics and is within the budget is selected for the prototype circuit. The prototype circuit is then constructed and tested to determine if all parameters meet specifications. Components may be replaced accordingly to the test results. Once all the components are selected and the circuit prototype meets the specification goals, the circuit is transferred to manufacturing.

Problems in selecting the passive components for the circuit design

One issue facing the equipment manufacturer circuit designer is the effective selection of passive components. Because of the wide variety of passive components, some circuit designers do not know how to choose or which to use. It is often the case that the circuit designer wants to just see the relative difference of similar components in terms of the circuit's performance. EDA tools are usually used to verify if the chosen component allows the circuit to meet its required performance.

The data that is provided by the component manufacturers is often applicable only at certain conditions. Therefore, circuit designers have to measure the component at the respective operating conditions such as at particular frequencies, bias level or temperature of the wireless handset.

However, the test setup for the measurement of a passive component at various conditions is complex and is time intensive with a network analyzer. In addition, measurement repeatability of high Q devices is questionable when using a network analyzer.

Another issue facing the equipment manufacturer is the replacement of obsolete components. In today's fast paced world, the product life cycles of components have become shorter. For example, for electronic equipment that is manufactured over a few years, the passive components within the equipment may become obsolete. The manufacturing engineer is then faced with the task of replacing the passive component. The manufacturing engineer must replace the passive component without changing the circuit's performance in the electronic equipment. Here again, the selection of passive components and the evaluation at the equipment's operating conditions is necessary.

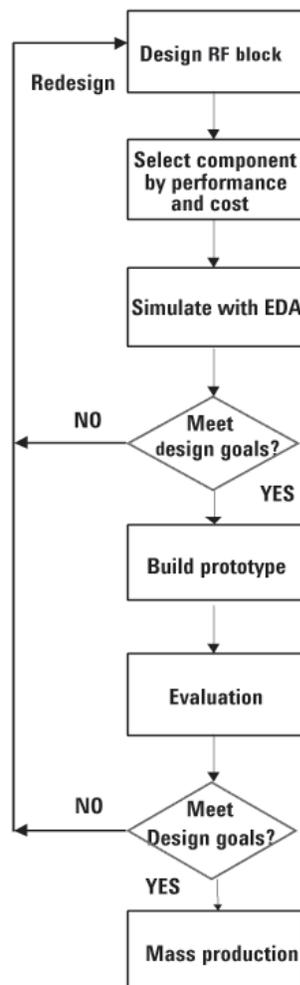


Figure 15. Design phase of the equipment manufacturer circuit designer

Using the E4991A and EDA tools for the effective selection of passive components

Before using the EDA tool for the selection of passive components, it is highly recommended to take the appropriate impedance characteristic data with the E4991A rather than a network analyzer. First of all the E4991A is capable of sweeping in various parameters such as frequency signal level and DC bias level¹ (voltage or current), enabling the evaluation of signal level dependent components such as varactor and PIN diodes. Since there is no need for a complex setup or a significant expenditure of time, data can be taken and imported into EDA tools in little time. In addition, repeatable data for high Q devices such as multi-layer ceramic capacitors (MLCC) and inductors can be obtained without any difficulty (See appendix for additional information).

Selection of a varactor diode in a 2.4 GHz RF VCO

Here, an example of selecting the appropriate varactor diode in a RF VCO is introduced. The ADS schematic of this RF VCO is shown in Figure 16.

This RF VCO is obtained by using a fixed capacitor and a varactor diode in series with an inductor between the base and the ground of a bipolar transistor. This sub-circuit is the resonant oscillator circuit deciding the frequency of oscillation. The oscillation frequency of this resonator circuit is 2.4 GHz.

The measure of how much reactive energy is stored in the resonant circuit is determined by the quality factor Q. Since the varactor diode has a smaller Q than the SMD inductor and MLCC at 2.4 GHz, it becomes the determining factor for the resonant circuit's Q.

The resonant circuit's Q has a significant consequence for phase noise in an actual VCO. The relationship between Q and the VCO's phase noise (ξ) is shown below by the Leeson's equation:

$$\xi(f_m) = \frac{FkT}{2P} \left(1 + \frac{f_0^2}{4Q_L^2 f_m^2} \right)$$

F= noise figure of amplifier

P= feedback loop power

f_0 = oscillation frequency

f_m = offset frequency

Q_L = loaded (resonant circuit's) Q

Notice that an increase in Q_L is one effective way to lower the phase noise components. However, an increase in Q_L leads to more loss in the feedback path resulting in a lower P. Consequently, the selection of the varactor diode must be determined by verifying the phase noise and output power of the VCO.

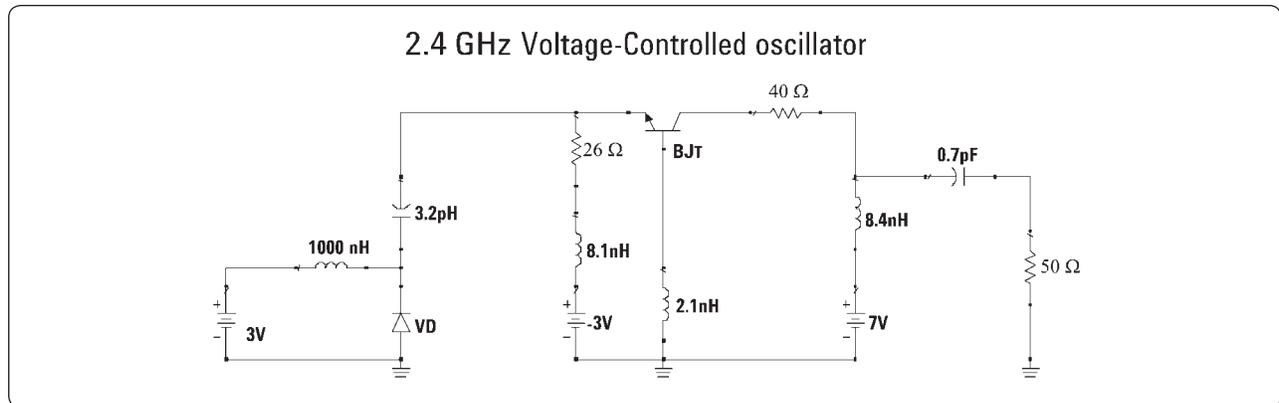


Figure 16. Circuit for 2.4 GHz RF VCO

1. E4991A option 001 is required.

In Figure 17, the circuit used for the selection of the varactor diode is shown. The impedance characteristic data for each of the varactor diodes were measured using the E4991A and the 16197A while applying 3 V DC bias. The measurement data (frequency sweep with fixed DC bias) was downloaded as a CITIFILE and imported into ADS as a Data Item component. By changing the file source of the Data Item component, two different varactor diodes' measurement data were used in the phase noise simulation. Notice that the measurement data for the SMD inductor and capacitor was also imported into ADS for better simulation accuracy.

The phase noise simulation results using two different varactor diodes (which have similar capacitance values at 3 V DC bias and 2.4 GHz) are shown in Figure 18. Whereas the output power has not changed, the phase noise is lowered by approximately 2 dB, when the varactor diode from company A is used in the circuit. In essence, using the varactor diode from company A, which has a better Q (See Figure 18) than the one from company B, helped in increasing the resonant circuit's Q.

As a result of this simulation, the circuit designer can now use the varactor diode from company A to evaluate the VCO's frequency vs. tuning voltage characteristics.

Conclusion

In this product note, the methods of how to simulate equivalent circuit models and how to effectively select passive components by utilizing the E4991A and an EDA tool such as ADS were introduced. The first discussion covered the design phase of the component manufacturer R&D engineer and how the simulation of equivalent circuit models using the E4991A and ADS can facilitate the design time for the passive component under development. An inductor's equivalent circuit model was used as an example.

The latter discussion covered the design phase of the equipment manufacturer circuit designer and how the simulation of circuit blocks using the E4991A and ADS can help in the selection of better passive components. The selection of varactor diodes for a 2.4 GHz RF VCO was used as an example. By using this product note as a reference, various needs in similar applications can be easily achieved.

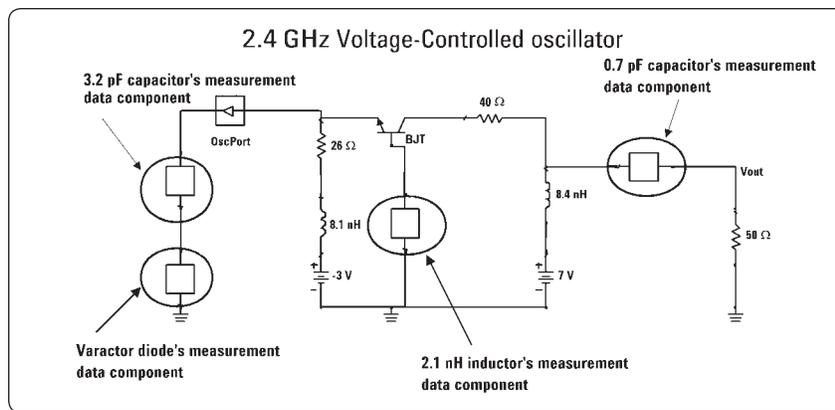


Figure 17. Simulated circuit in ADS

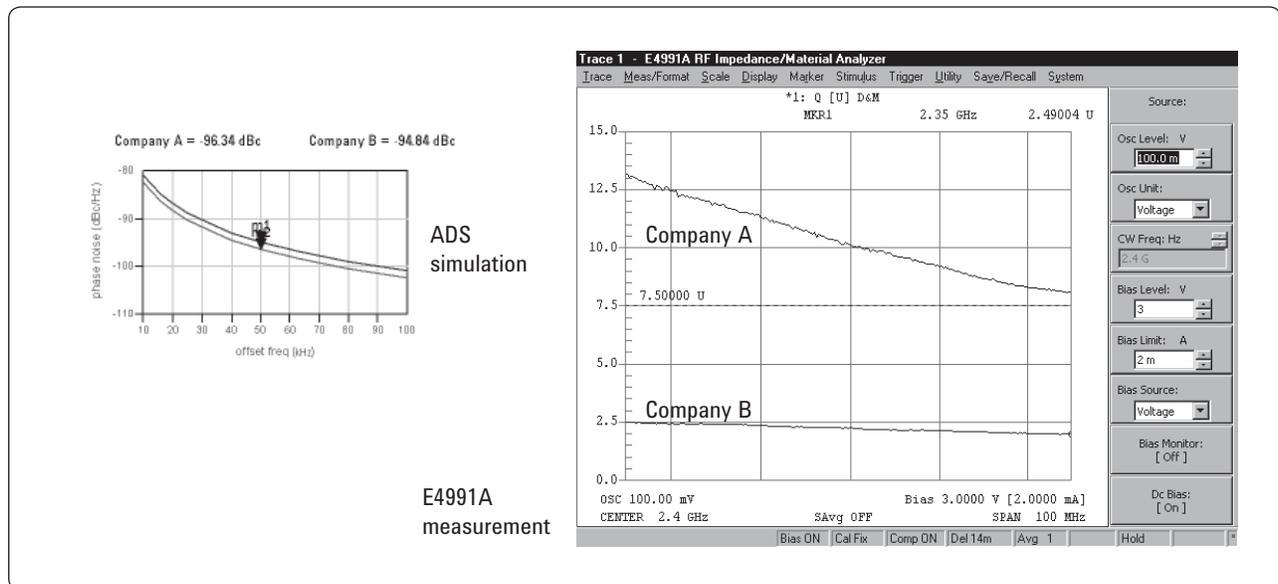


Figure 18. Phase noise simulation and Q measurement results

Appendix 1

In this section, measurement tradeoffs between a network analyzer and an impedance analyzer is discussed. Also, the tradeoffs from a circuit designer's point of view will be discussed.

Measurement repeatability

Stability in repetitive measurements is a key performance required for accurately determining the impedance characteristics of passive components. The variance of the measurement stability when a 1 nH inductor and a 1 pF capacitor is measured is shown in Figures 19 and 20. Notice that the results obtained by the E4991A demonstrate superior (low variance) measurement repeatability when compared to a best performance vector network analyzer in both low impedance and high impedance measurements.

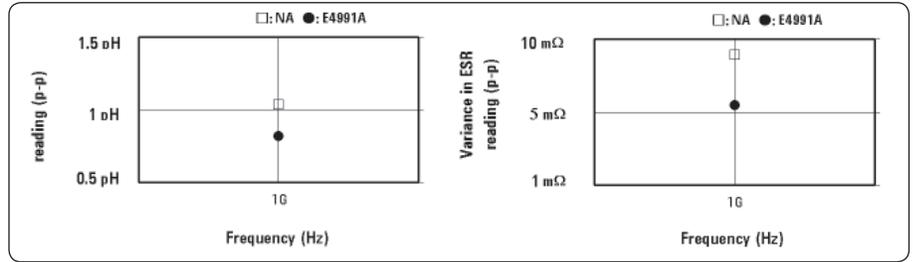


Figure 19. Variance in measurement data for a 1nH inductor

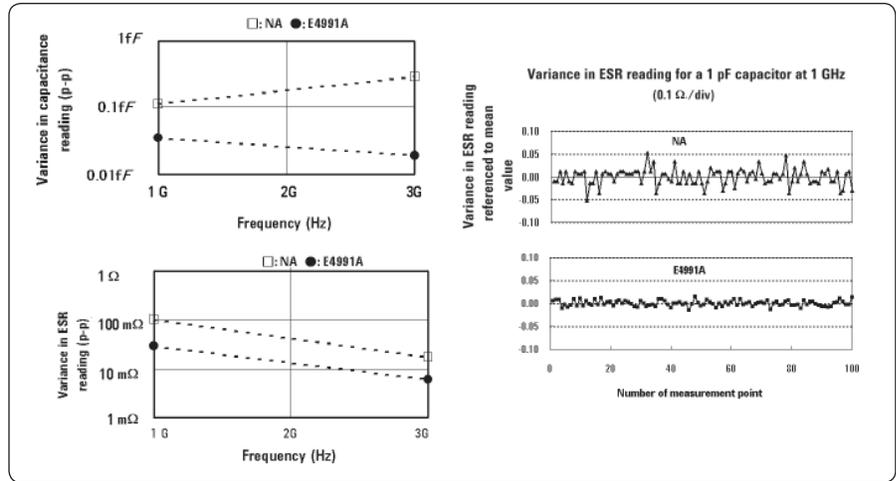


Figure 20. Variance in measurement data for a 1pF capacitor

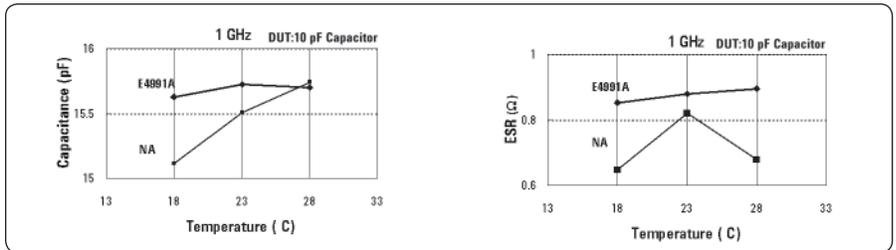


Figure 21. Temperature stability data for a 10pF capacitor

Note for Figures 19 and 20: Measurements were continuously repeated 100 times for each DUT without removing it from the fixture. The 16196A was used for both the network analyzer and E4991A to make these 1-port measurements.

Note for Figure 21: The 16196A was used for both the network analyzer and the E4991A to make these 1-port measurements.

Figure 21 shows the temperature dependence test results of a 10 pF capacitor measured at 1 GHz. Since the temperature coefficient for this capacitor is specified as less than 30 ppm/°C, the capacitance and ESR values are not supposed to show signs of change as the temperature is varied. Notice that the results obtained by the E4991A exhibit stability in temperature variations when compared to the network analyzer.

For more information on the details of this repeatability analysis, refer to *Advanced impedance measurement capability of the RF I-V method compared to network analysis* literature number 5988-0728EN.

Difference from a circuit designer's point of view

As shown in Table 1, the advantage of using the network analyzer instead of the impedance analyzer when checking the relative difference of passive components is that the circuit board's characteristics can be included with the component's characteristics.

This is much closer to the real world because the component's mounting conditions on the actual circuit board of the electronic equipment can be measured in a 2-port configuration. An impedance analyzer cannot measure the mounting conditions since it can only do 1-port measurements.

Table 1. Tradeoffs from a circuit designer's point of view

	Advantages	Disadvantages
Network Analyzer	Can take component's data with the PCB's effect.	No standard fixture is available. Must mount component on PCB.
	1-port and 2-port measurements	Inferior impedance measurement repeatability and temperature stability
Impedance Analyzer	Well-characterized test fixtures are available	PCB's effect cannot be taken into account
	Superior impedance measurement repeatability and temperature stability	1-port measurements only

This difference is critical for a circuit designer. Because the circuit designer must select components when designing the circuit block. If a measurement configuration consisting of the network analyzer is used, passive components must be manually mounted on the PCB (printed circuit board) before measurement. Each time the component is mounted a different characteristic will be measured since the positioning of the component greatly affects the inductance or the capacitance that is being measured. As a result of the measurement repeatability of the network analyzer combined with the positioning repeatability of the component on the PCB, only a low repeatability can be attained.

If on the other hand, the E4991A is used, since the 16196A/B/C/D and the 16197A have high positioning repeatability and the E4991A's impedance measurement repeatability and temperature stability is high, a high repeatability can be attained. As a result, by using the E4991A, the difference between the quality of passive components can be easily observed.

Appendix 2

This section explains how to use the 4294A precision impedance analyzer for device modeling using the 4294A precision impedance analyzer.

Superior impedance measurement accuracy

The 4294A precision impedance analyzer achieves a wide impedance measurement range and superior measurement accuracy with by adopting the auto balancing bridge (ABB) and the four-terminal pair (4TP) configurations (Figure 22).

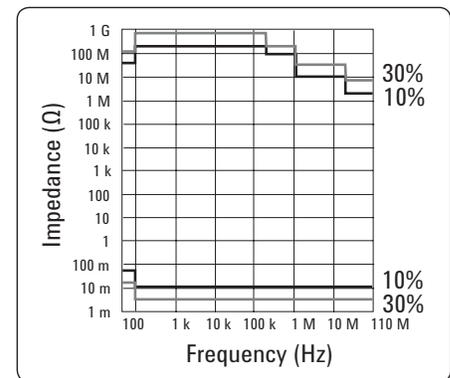


Figure 22. Impedance measurement range of 4294A (typical)

SMD test fixtures provide excellent measurement repeatability. The 16034G/H is available as an SMD test fixture for the 4294A. This test fixture covers the 110 MHz frequency range and the applicable SMD size.



Figure 23. 16034G/H SMD test fixture

Linkage integration with EDA tools

The 4294A can convert impedance measurement data into S-parameters, that can be saved in the Touchstone format¹. The data can be transferred to the a PC by using a LAN or by saving it on to a floppy disk.

When using the 4294A for device modeling, the modeling circuit used with the E4991A can be reused with the 4294A by changing the data format setup. Changing this setup is necessary because the E4991A supports the CITIFILE data format while the 4294A supports the Touchstone format. Once this step is done, allowing you to import the S-parameters into the data item component, the rest of the procedure is the same as that of the E4991A.

References

1. *Impedance Measurement Handbook*, Agilent literature number 5950-3000
2. Application Note M024, *Voltage-Controlled Oscillators Evaluated for System Design*, Agilent literature number 5963-3222E
3. Application Note 1034, *Designing VCOs and Buffers Using the UPA family of Dual Transistors*, California Eastern Laboratories
4. Application Note 1016, *A Low Noise VCO Design for PCS Handset Applications*, Alpha Industries, Inc.
5. Document 158-1, *Modeling Coilcraft RF Inductors*, Coilcraft
6. *TDK S-Parameter Data Library*, TDK Corporation Electronic Components Business Group

Web Resources

Please visit our web site at:

www.agilent.com/find/impedance

for more information about impedance test solutions;

www.agilent.com/find/eesof

for more information about Agilent EEsof EDA.



Agilent Email Updates

www.agilent.com/find/emailupdates
Get the latest information on the products and applications you select.



Agilent Direct

www.agilent.com/find/agilentdirect
Quickly choose and use your test equipment solutions with confidence.



www.agilent.com/find/open

Agilent Open simplifies the process of connecting and programming test systems to help engineers design, validate and manufacture electronic products. Agilent offers open connectivity for a broad range of system-ready instruments, open industry software, PC-standard I/O and global support, which are combined to more easily integrate test system development.



www.lxistandard.org

LXI is the LAN-based successor to GPIB, providing faster, more efficient connectivity. Agilent is a founding member of the LXI consortium.

www.agilent.com

For more information on Agilent Technologies' products, applications or services, please contact your local Agilent office. The complete list is available at:

www.agilent.com/find/contactus

Americas

Canada	(877) 894-4414
Latin America	305 269 7500
United States	(800) 829-4444

Asia Pacific

Australia	1 800 629 485
China	800 810 0189
Hong Kong	800 938 693
India	1 800 112 929
Japan	0120 (421) 345
Korea	080 769 0800
Malaysia	1 800 888 848
Singapore	1 800 375 8100
Taiwan	0800 047 866
Thailand	1 800 226 008

Europe & Middle East

Austria	01 36027 71571
Belgium	32 (0) 2 404 93 40
Denmark	45 70 13 15 15
Finland	358 (0) 10 855 2100
France	0825 010 700*
	*0.125 €/minute
Germany	07031 464 6333
Ireland	1890 924 204
Israel	972-3-9288-504/544
Italy	39 02 92 60 8484
Netherlands	31 (0) 20 547 2111
Spain	34 (91) 631 3300
Sweden	0200-88 22 55
Switzerland	0800 80 53 53
United Kingdom	44 (0) 118 9276201

Other European Countries:

www.agilent.com/find/contactus

Revised: October 6, 2008

Product specifications and descriptions in this document subject to change without notice.

© Agilent Technologies, Inc. 2001, 2004, 2008
Printed in USA, November 13, 2008
5988-3029EN

1. This function requires firmware rev. 1.10 or later.



Agilent Technologies